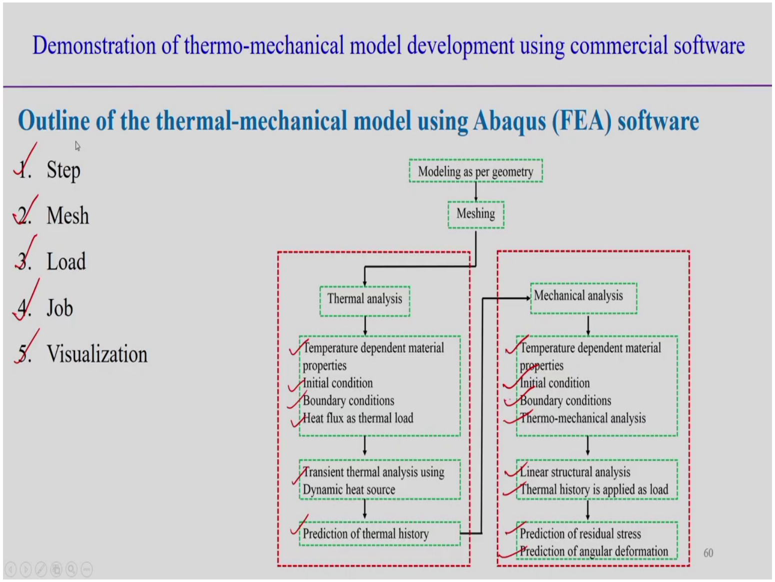
Finite Element modeling of Welding processes
Prof. Swarup Bag
Department of Mechanical Engineering
Indian Institute of Technology, Guwahati

Lecture - 32
Demonstration of Thermo-mechanical model development using commercial software

(Refer Slide Time: 00:32)



Hello everybody. So, far we have discussed the thermo mechanical analysis, what way we can develop a particular model and that overview of this mechanical analysis; and that what way we can integrate the thermal analysis and the mechanical analysis in a particular welding process.

Now, we will try to look into that whether this thermo mechanical analysis is possible or not by using some kind of the commercial software. So, this particular sub section we will focus on these things using the this commercial software. We will try to look into that how this thermo mechanical model can be developed.

So, to look into this if we first if we look into that first step is that we have to create the geometry. That means domain, domain of the analysis basically the physical domain of the analysis. So, there are several way because it can be the multi domain also. Because, you can define some for example, in case of arc welding process so, then creation of the arc and then the shielding gas flow rate that can be the one domain of analysis.

But, normally in general the simplified way we can represents this any kind of the welding process, in the form of a mathematical model like that if you focus on the analysis only on the workpiece surface. Because finally, the two plates will be joined together so, basically in this case these plates, we are supposed to join this together that is the solution domain.

So, therefore, we consider this as a geometry and within this the application of the governing equation boundary condition this particular domain will be able to get the output in the form of a temperature distribution as well as distribution, stress, all these can be kind of parameters or output solution we will be able to get from a finite element based software.

So, in general first step is that once we decide the model geometry, then meshing is also required meshing means the discretization of the domain in the particular that creation of the large number of elements and as well as the nodes.

So, therefore, after discretize the domain in case of thermo mechanical energy first we focus on the thermal analysis. So, in this case thermal analysis once you focus the thermal analysis, then it is this is a first step will be though define the temperature dependent properties first thing.

Second is the initial condition initial condition is required, if we solve the transient problem. But, if we solve the steady state problem then we would not require any kind of the initial condition, but both the cases we need the boundary condition. So, once we define the boundary condition in a thermal analysis that we have already explained in the schematically that what are the boundary condition, typically in case of the thermal analysis model.

Then, after that we define either heat source model or simple heat flux through the boundary we can define. And, then this is the property is the initial condition, boundary condition and for thermal analysis you have to choose the governing equation also and normally in thermal analysis, we solve the Fourier heat conduction equation. So, defining all these particular or choosing this particular option, then we can perform the transient thermal analysis using the dynamic heat source model.

So, transient thermal analysis using the dynamic heat source model means that heat source model can be dynamic in nature also for example, in case of pulse also. There is a continuously on off on off mode; that means, during the pulse on fluid, we are supplying the energy the flux to the domain, but during the off period we do not supply the energy to the domain.

So, that is way we can define the heat source model and accordingly we can do the transient thermal analysis, or if necessary we can do also steady state thermal analysis as well. Then, after completion this thing this is the then predict the thermal history that is the most important predict the thermal history means, then we will be able to know what is the time versus temperature diagram particular point or in the whole solution domain what is the temperature distribution also in particular instant of time.

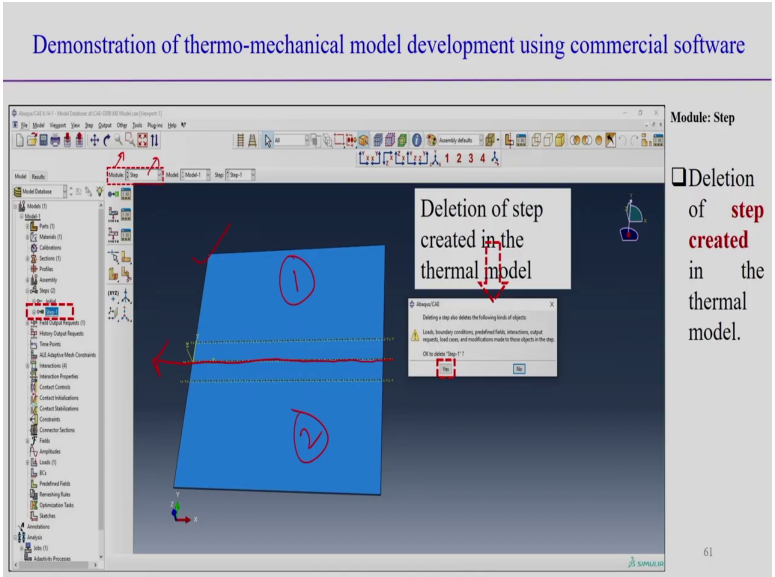
So, then that kind of the output we can get from the thermal analysis. And this output can be considered as a input to the stress analysis model, or we can say the mechanical analysis model. So, even similarly way once we decide the mechanical analysis, then we have to define the temperature dependent property same initial condition also.

If and then boundary condition and then we perform the thermo mechanical analysis. So, once we consider the thermo mechanical analysis only the thermal analysis as a input to the mechanical model. Then, we can do the linear structural analysis and in this case the thermal history is applied as a load for the mechanical analysis. So, then it predicts the residual stress distribution, prediction of angular distribution, normal distribution and from there we can get the several form of the results also.

But finally, we are interested to know what is the residual stress and distortion in a welded structure so, that is the overall view or overall procedure for any kind of thermo mechanical analysis by using any commercial software. But, this particular software when we are using commercial software, there are several steps basically step itself is that first is the step in a commercial software, then next will be the mesh this how we can apply the load and maybe first step we can how we decide the steps basically.

And then finally, job application and then visualization of the once of the solution complete. Then, we will look in the visualization of the result these are the very basic steps, thermo mechanical analysis in any using any kind of the commercial finite element based commercial software.

(Refer Slide Time: 05:47)



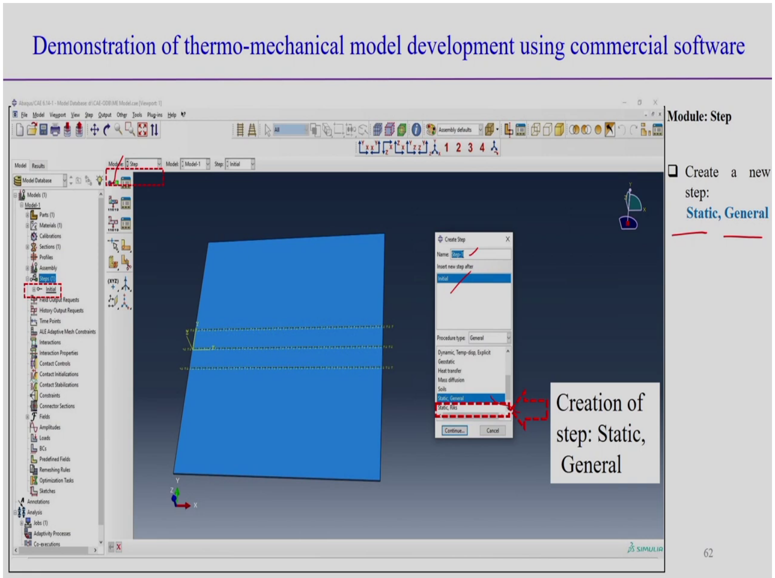
Now, if we look into these steps also if we see remember the module, we decide in the first step in the module. If we just is highlighted here the module here you can decide the step. And step first is the we can decide the model as a model 1 and step equal to step up 1. So, decide and in this case deletion of the step created in the thermal load.

So, basically we have to remove the steps what we have created using the thermal load, because this is stress analysis we can use the same thing. So, deletion of the step create an thermal model first we can do these things and here, we can see this step indicates and this is the here this figure indicates the solution geometry.

This highlighted part is there also that some once we create the solution geometry for example, this is the one side and plate 1 and plate 2 we are joining in a butt joint configuration. So, at the center the heat source moves along the center and this thing along this direction and this highlight part is there is a some meshing strategy we can follow.

For example, very fine mesh we can create near about the application of the heat in that particular zone or maybe we consider the fusion zone and heat effective zone, but IAF from the world center, we can use the some course mesh also to reduce the computational time. So, these are the first step is we understand that the first deletion of the step created in the thermal model, this is the first part of a thermo mechanical model.

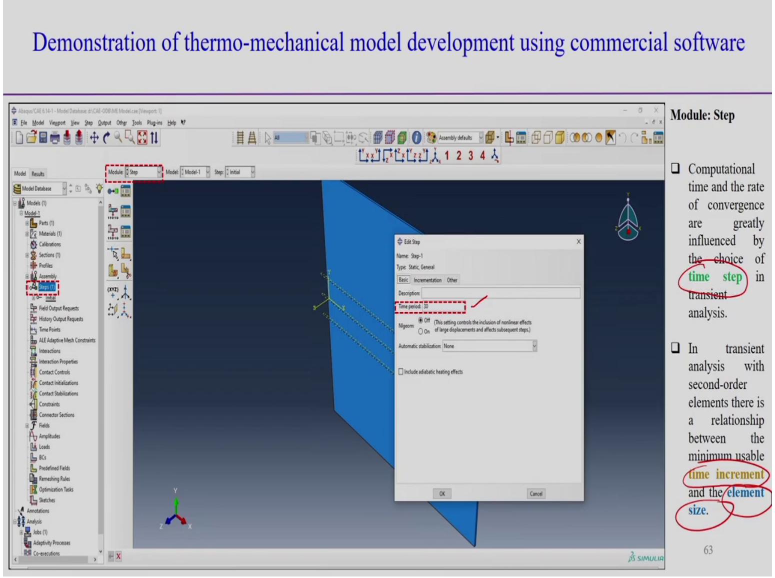
(Refer Slide Time: 07:18)



Next, we can see create a new step for the mechanical analysis. And, then and mechanical analysis follow the static analysis and in general. For example, in this case we can see here also, we can from this option it is a static general and here we can then name the general create the step we can put some name of the step.

And, then inside the new step here we can see the initial select these things, and here we have selected that static creation of the step in the static general step. That means, static that equilibrium equation force equilibrium equations following that we have shown in the theory part also, that what we normally follow in the stress analysis. So, therefore, that is the indication we use this step.

(Refer Slide Time: 08:01)



Once we decide the step and then we can see we have to decide the time period, for this analysis for example, the computational time and the rate of convergence are greatly influenced by the choice of the time. So, it is very important to decide the time step in the transient analysis what time step we can decide. And also the total computation time also depends on the this time step as well as the total time period depends on the actual welding condition.

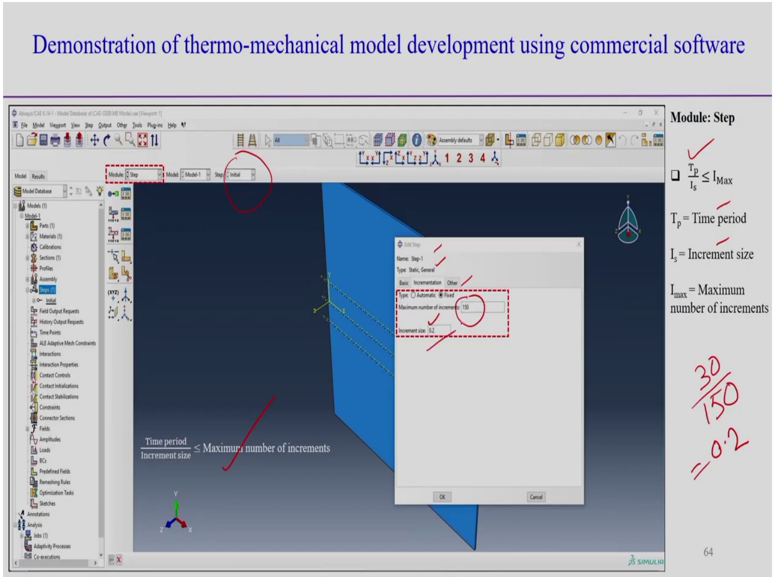
So, it means that what welding velocity we are considering, what is the length of the work piece we have considered, domain of the analysis accordingly we can decide what is the total time required to travel from one end the another end. From that point we can decide the total time step. So, in these cases we have used for example, time step is 30, but the units is already very we have to define the units before hand before start of the analysis, we can set the what kind of units we are using for this particular analysis.

For example, if we use the SI system. So, accordingly length time will be considered according to the SI system. So, that has be defined before the analysis starts. Now, in transient analysis with second order elements, there is a relationship between the minimum usable time increment and the element size.

So, that is also important the what may be time step the time increment and the element size they are having some relation and specifically that is true for. In case of the second order elements or even first order element also. If we decide that what should be the time step, there may be some relation with the what are the mesh size will normally link with the mesh size and the properties also material properties.

So, the by looking into the stable time step; that means, maximum time step we can decide the time step that less than that of the maximum time step. So, that is decided on the what is the element size geometry of the particular element and then it depends on the order of the elements we are considering for a particular analysis. So, that is the step we decide the total time as well as the we can choose the time step also that is the next step.

(Refer Slide Time: 09:55)



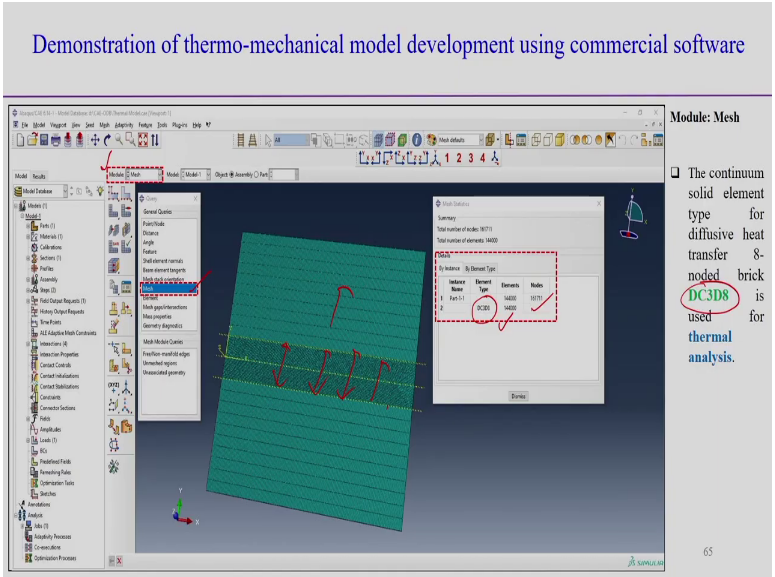
Further step we can see for example, here in this particular module and particular model and particular step, step you can see in this case we have decide the step; step is indicated at the initial step. So, therefore, in this particular software, this is the initial step and the time we can consider the step and the see this particular the time period and the increment size. What is be the increment size? I max is the maximum number of increments.

So, here you can decide what is the maximum number of increments 150. So, therefore, if we decide the total time is 30 and maximum number of increment so, that we can say as a single step total number of steps equal to 150. So, therefore, increment size this equal to 0.2 that is the value. So, that indicates that this is the increment size.

So, we have to decide the number of increments. So, that we can decide the automatic also, then Abacus will choose according to their scheme or algorithm the automatic the increment size can be decided or we can use the fixed or fixed value also. And that once we choose the fixed value the increment size for each step, what should be the increment size can be decided.

And this also we can see the ratio that maximum number of iteration can be this ratio, T p minus I s increment size time period by increment based on that we can decide. So; that means, it decide the maximum number of increments here in this particular selection. So, in this case, the step 1 and initial step name equal to step 1, you can see the type equal to static general. And, then with the fixed time step and here maximum number of increment and increment size is 0.2 is calculated by this particular software.

(Refer Slide Time: 11:33)



So, this is the next step now once we decide the times step size also, then we can look into the mesh what way we can generate the mesh. So, we can see this mesh can be decided that next step is the mesh where we can see that module here, which is simply the module is a mesh here we can select the module and the mesh.

So, in this side the continuum solid element type so, for the diffusive heat transport we normally need 8 noded brick. This element is used for the thermal analysis DC3D8 actually there are element properties also. So, it depends on the particular commercial software and there are so, many different elements or types of elements are available.

And, then one particular element may be useful one particular analysis for example, in case of thermal analysis we can choose a particular element. So, therefore, we have some idea about the properties of the element and that is maybe associated with any kind of the commercial software before choosing this particular element.

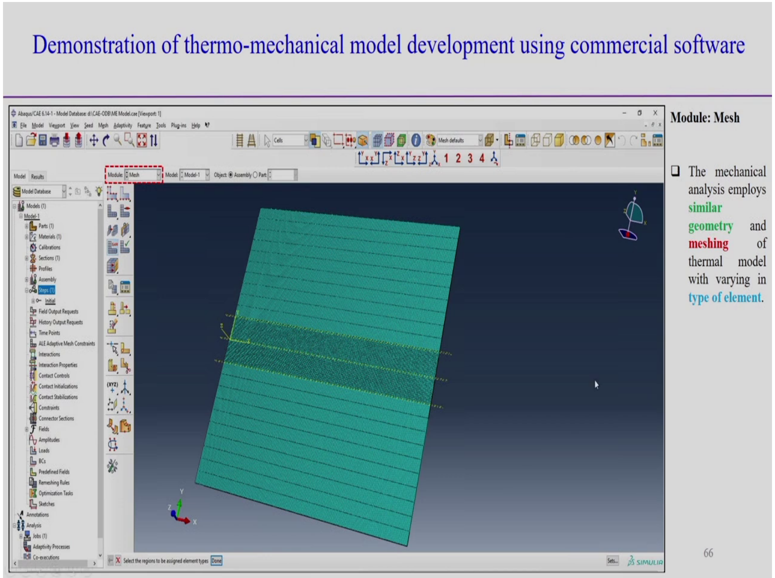
But, for this particular commercial software we use this DC3D8 this particular type of the element we normally choose for the thermal analysis, but it is not necessary that same element will may be useful in case of the stress analysis. Stress analysis the element type of the element can be different according to the nature or of the element. So, there may be some least of the type different types of the elements and which elements is suitable what kind of the analysis.

So, that prior knowledge of this particular type of the element is required. So, once we do this thing then we can choose some mesh also, we can choose automatic meshing also or we can choose assuming some linear brick elements. So, then once we decide this thing if we look into that the near about the weld central line, it creates the very fine mesh away from this thing the mesh size.

It is a coarse mesh we normally use, here we can some idea the total number of elements is this one 144000 is the total number of elements in this particular analysis. So, therefore, total number of nodes will be 161711. So, this way we can discretize the domain, by choosing the particular element type.

And then here you can see the element type is used and total elements is this one and total nodes the calculated that kind of information we can get. But, accordingly we can decide the mesh. So, maybe this is the stepping of the mesh creation of the mesh.

(Refer Slide Time: 13:44)

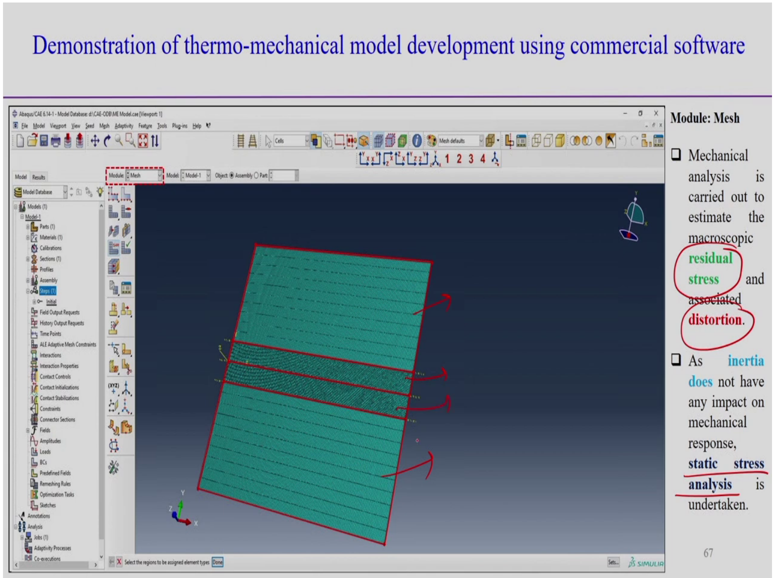


Now, in this case module as a mesh in this particular commercial software here we can see the mechanical analysis employs the similar geometry, and meshing of the thermal model with the varying in the type of element. So, therefore, similar geometry for thermal analysis we have shown this is the we create the element and all this thing.

The same kind of the meshing we can use for the stress analysis also, but type of element can be different in case of the stress analysis. It is also in other point of view maybe we can say that even in thermal analysis as well as the stress analysis also, it is better to use the same kind of the meshing.

So, then it will be useful to use that data because thermal analysis what the each and every node point, we normally store the temperature data and that same temperature data we use for the stress analysis. So, it is better to we can choose the similar geometric for the thermal analysis as well as the mechanical analysis, but type of element can be different.

(Refer Slide Time: 14:37)

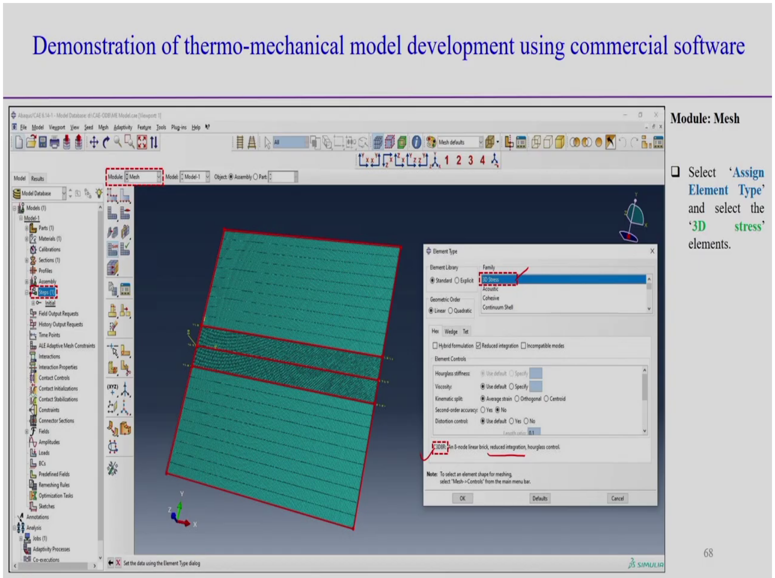


Here we can see even with the module mesh in this particular module. The mechanical analysis is considered to estimate the microscopy residual stress and the associated distortion. So, definitely that is the purpose of using the mechanical analysis the residual stress and distortion is the objective to do to perform the mechanical analysis.

So, therefore, inertia does not any impact on the mechanical response say for static stress analysis, we normally consider is undertaken in this particular process. So, this is a very simple analysis we consider the static analysis and we are not considering any inertia effect during this mechanical analysis.

So, accordingly we can choose this thing, here we can select the this different domain. Here you can see the in this particular meshing also in the geometry that these are the different domain, one particular type of mesh we are followed course mesh and these are the domain here we can follow the fine mesh also.

(Refer Slide Time: 15:24)

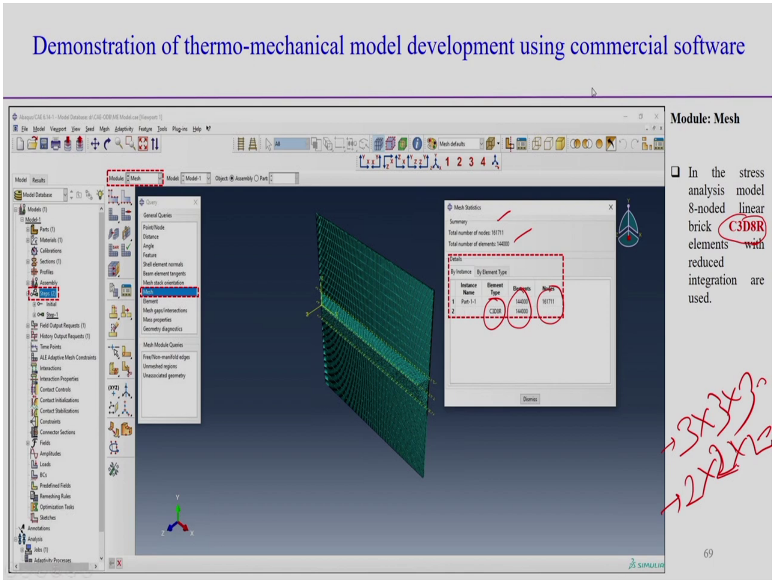


Now, module mesh also and then select assign element type what type of element we can choose for in case of the stress analysis, but here we can see some idea also in the stress analysis part what we can choose the this kind of the 3D stress. So, that is in element library we have the different kind of the analysis module.

So, therefore, in this since we are doing the stress analysis we consider the 3D stress element. So, we choose we highlight this 3 the stress element and accordingly this here, we can give the information that C3DR8, these are the particular type of the element we can use for the stress analysis.

And, we can see this particular type of the element is associated its indication of the 8 node linear brick element. And reduce integration scheme is used and our glass contour also possible by this particular type of the element. So, therefore, some sort of analysis what kind of analysis, we are doing accordingly we can choose the type of the element which is available in the element library in any kind of the commercial software.

(Refer Slide Time: 16:24)



Now, here you can see the mesh statistics also in the stress analysis model. The linear brick element C3DR8 with this particular element we can choose. And, here we can use the reduced integration reduced integration scheme is normally useful, because in certain part of element when we do the elemental matrix formation.

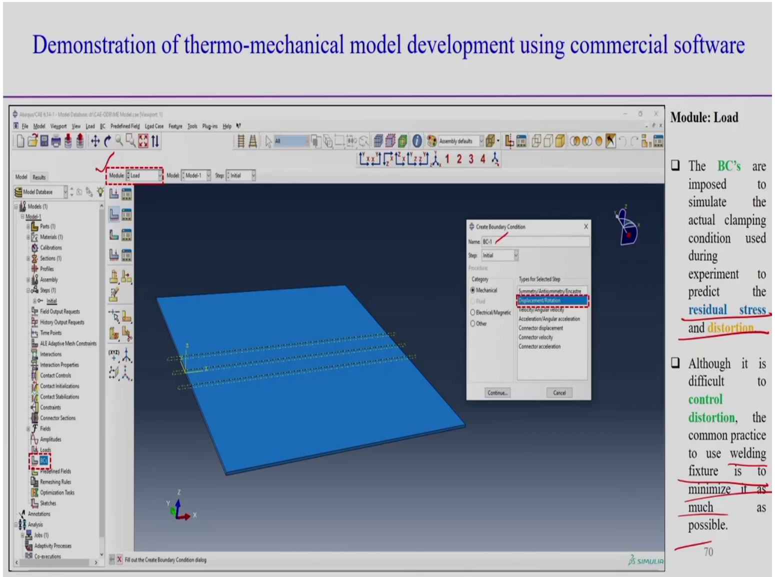
Once you follow the elemental matrix formation that we have sawn when the basic, when you look into this particular element and then we estimate the integrate over this domain one particular element. And, then if we follow particular integration rule also that means, if we follow 3 by 3 if you remember the 27 integration points if we consider also.

And some cases we can use the 2 by 2 in 3D analysis 2 by 2 integration total 8 integration points in certain cases also, we can use the 27 integration points to find out the volume integral in numerically in a particular element. So, therefore, these some cases use these number of points integration points within this particular domain.

And some cases we use this less number of integration points in particular. So, in this case 2 into 2 into 2 that number of integration first indicates the reduced integration and this may be useful in. Because, in certain cases or type of the analysis or order of the equation, say then accordingly we can choose the whether there is a need of reduced integration scheme or not and depends on the other parameter.

So, maybe sometimes the locking of the particular element so, to avoid that kind of situation normally we use, the reduced integration scheme in these cases. So, therefore, in stress analysis model we can see in general that this element type chosen total number of element, total number of nodes. And we can see in summary the total number of nodes this kind of information, we can guess the mesh statistics in a particular commercial software so, after choosing the type of the element and meshing then type of the element.

(Refer Slide Time: 18:14)

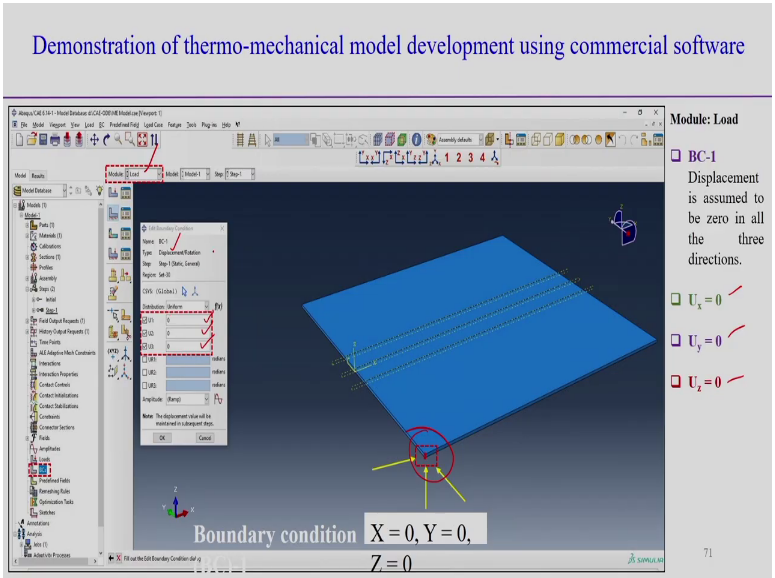


Then, next step we can choose the load condition; that means, this if you see the next module is the load, here we can choose the loading means what way we can apply the boundary interaction or thermal load or in this particular case or. So, load means the boundary conditions imposed to simulate the actual clamping condition used during the experiment to predict the residual stress and distortion.

Its it means that in this case the displacement or rotation both kind of the boundary interaction can be done in the mechanical analysis. And you can see it indicates the BC 1. So, basically boundary condition 1 and this boundary condition once decided what are the clamping force we can use the during the actual welding process the same thing we will try to represent in the mathematical form the or we can put the boundary condition.

We can show that what way we can we can put the boundary condition, all that is difficult to control distortion, but the common practice to use the welding fixture. We normally use the welding fixture is to minimize it as much as possible. So, definitely it is very difficult to control the distortion. So, we normally control to some extent the distortion using the particular design of the welding fixture, but what way we can implement the boundary condition also in this particular analysis.

(Refer Slide Time: 19:27)



Here we can see that imposing of the boundary condition here, this imposing of the boundary condition comes under the module of the load. Here, if we choose the load under the module, then we choose the decide the boundary condition and, this boundary condition in terms of the boundary condition in terms of the displacement also as well as the rotation also can be used.

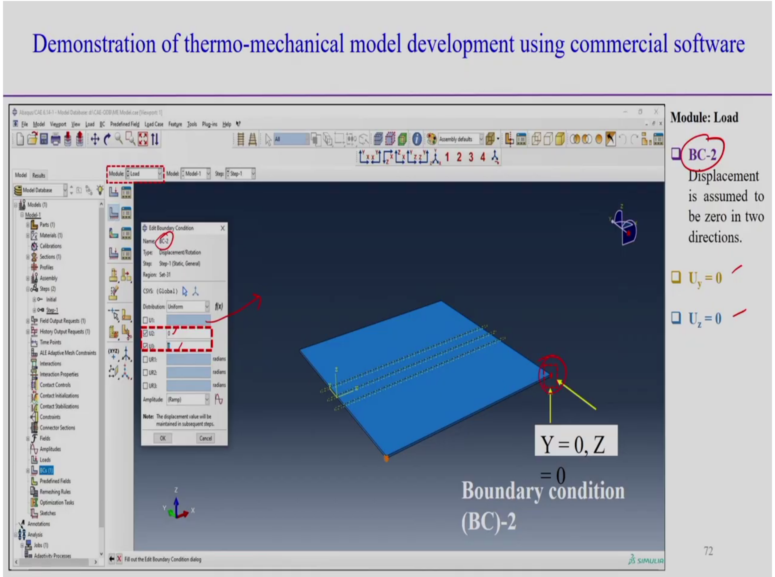
So, in this particular problem we use the displacement which assumed to be 0 in all the three direction for example, in this particular node point. So, this particular two nodes point due to corner points see these two corner points the clumping is there normally happen. So, this clumping is there then it actually restrict the distortion of a this particular node.

So, in this particular node we assume that all the displacement along X, Y and Z direction is 0 that is the representation of the boundary condition. So, X equal to 0, Y equal to 0, no displacement and Z equal to 0; in the other sense that sorry this is the point. And in this case the U x, U y and U z this is the displacement field. So, all in X, Y and Z direction these all are 0.

So, that we simply put this thing distribution uniform, then with the check in with the highlight the U 1 equal to 0, U 2 equal to 0 and U 3 equal to 0. So, this mean integral U x, U y and U z all are 0 and we are not imposing any kind of the rotation degrees of freedom. Even we are not considering the rotational degrees of freedom or we are not putting any boundary condition for the rotational degrees of freedom, simply we are considering the displacement degrees of freedom.

And that is the boundary condition 1. For example, these particular cases if we decide we put the boundary condition, we have to select the node. Then, accordingly we put the boundary constant using this particular option or boundary condition.

(Refer Slide Time: 21:11)

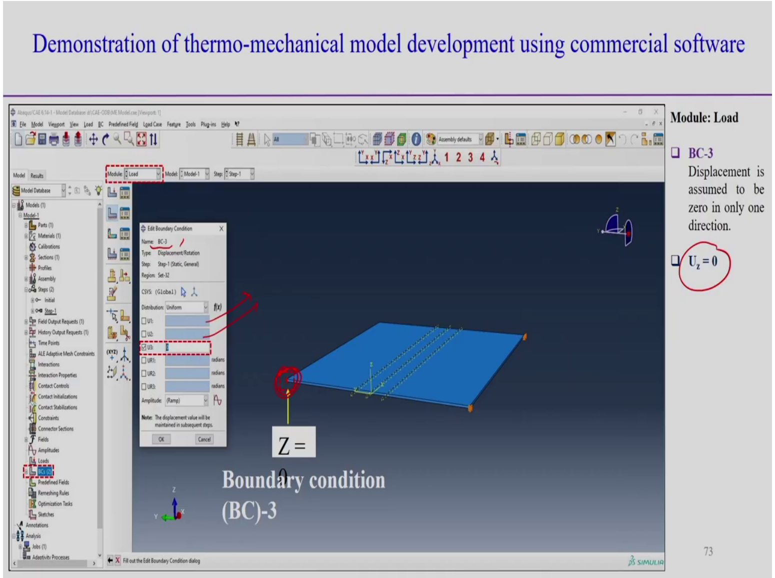


Now, maybe look into the other node also in this particular node the boundary condition 2 and then in this case here we can see that U y equal to 0 U z equal to 0. That means in this case the only 2 degrees of freedom we arrest it means that, you we are not imposing any boundary condition for U 1 So; that means, there may be displacement happen along the X direction. But imposing the boundary condition the displacement restrict the displacement along the Y and Z direction.

So, their position can be simply 0 in these both the cases. And degrees of freedom or the rotational degrees of freedom we are not arresting we are not putting any kind of the boundary condition not restrictive. So, this way if we physically decide that these are the boundary condition, according to the welding condition the same can be imposed in the form of, if we see this is the boundary condition 2 BC-2.

Here, we can see the BC-2 also and put the displacement boundary condition and in this particular step 1. And then this is the particular zone and here we are putting the boundary condition as the only the two arrest only 2 degrees of freedom.

(Refer Slide Time: 22:12)

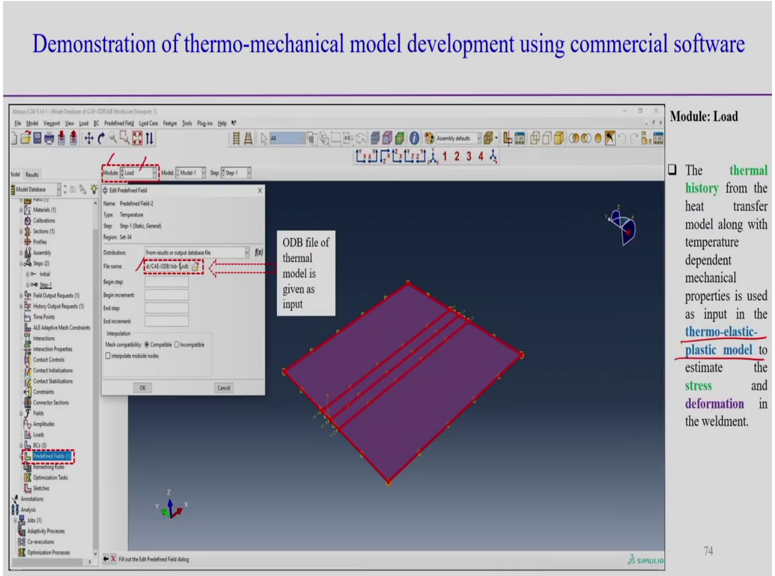


Similarly, we can show that if suppose you want to arrest only 1 degrees of freedom, in the other node also in this particular zone. So, boundary condition 3 here we can see displacement is assumed to be 0 in only one direction so, then only U z equal to 0. So, that is why U z equal to 0 if we put the boundary interaction here. So, this is as a boundary condition 3. So, displacement field and we see we are not choosing any kind of the boundary condition in X and Y direction.

But, only Z direction we are putting the boundary condition equal to 0. So, this way different type of the displacement boundary condition can be put, in case of the stress analysis model.

But, before that we have to decide which part we have to consider boundary condition, but if you restrict the all the corner of too many restriction was there, then the material becomes very stiff. And then the level of the residual stress generation can be very high.

(Refer Slide Time: 23:00)

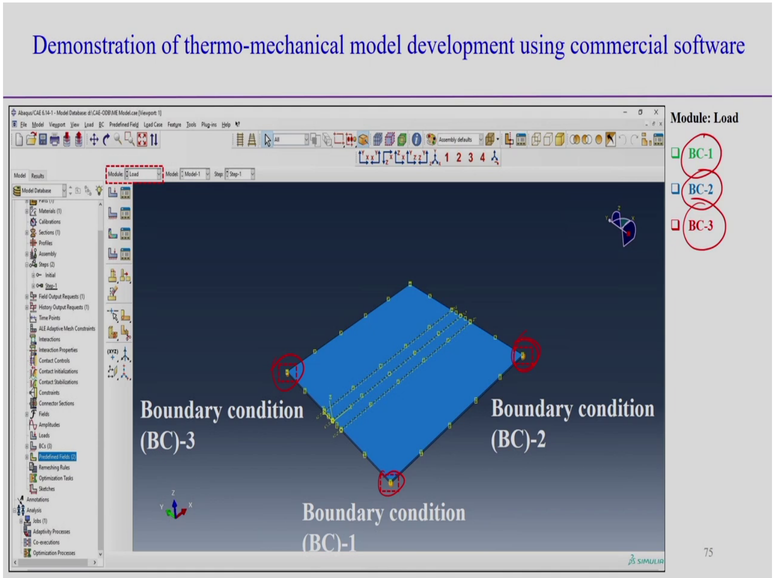


Now, you create the ODB file for the thermal model the imposing the ODB file so; that means, now we are putting the loading means first we decide the boundary condition, then we have to put the thermal loading to this stress analysis domain. So, that the thermal history from the heat transfer analysis along with the temperature dependent properties mechanical properties, is used as input to the thermo elastic plastic or we can say the stress analysis model.

And, then that will be giving the stress and distortion. So, in this particular case that module even we choose the same model load, but not the boundary condition rather we can choose the imposing the thermal load. That means, basically the data from the thermal analysis for the stress analysis model that is called the ODB file of the thermal model is given as the input.

And here you can see the ODB file we can choose the this ODB this file for the thermal model is given as a input, to this domain for the stress analysis model in this particular load step.

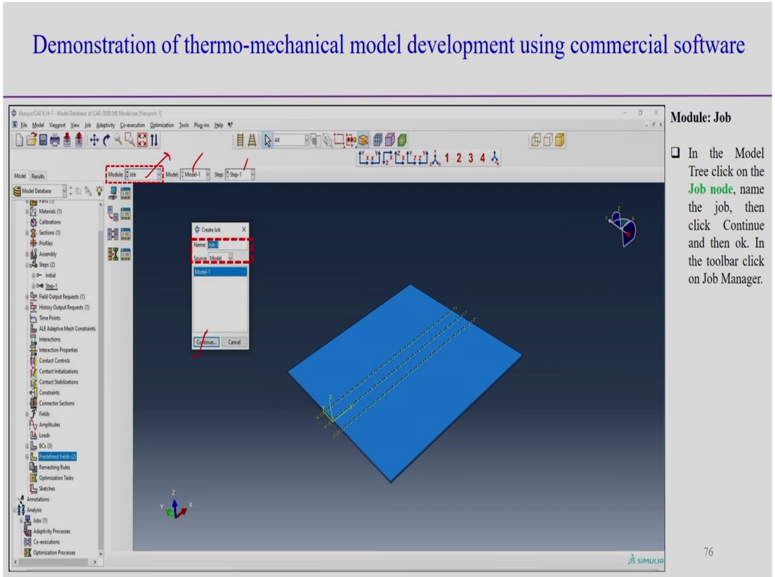
(Refer Slide Time: 24:00)



Now, we can see that module in the load step with the application of the thermal load, and then all the BC boundary conditions 1, boundary condition 2 and boundary condition 3 here, we can see the boundary condition 1, boundary condition sorry 1 boundary condition 2 and boundary condition 3 along with the thermal load.

Then this is the now we can say the before start of the mechanical analysis all the inputs to the given to the solution domain for the stress analysis model. It means that all the thermal load as well as the all boundary condition are already given, for the stress analysis model now it is ready to analyze for the mechanical analysis can be done.

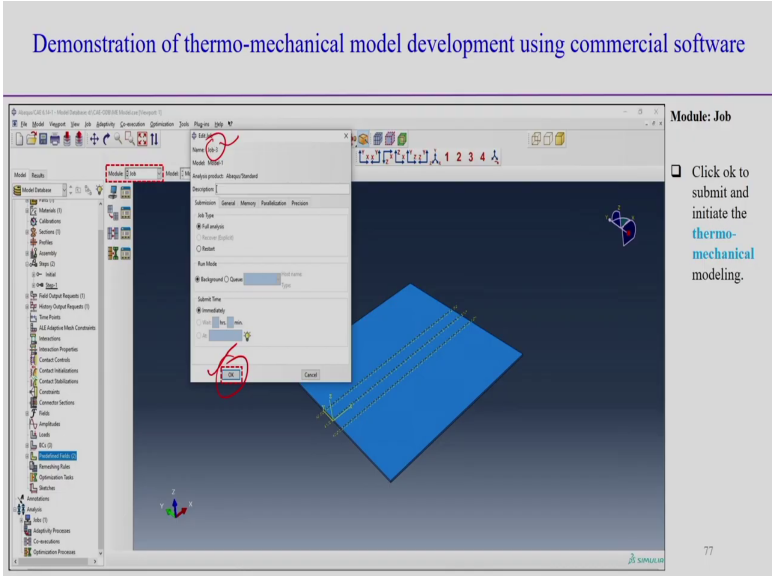
(Refer Slide Time: 24:40)



So, here we can put the module they change the module after load, then here the module becomes job. So, then a job for the model 1 and the step 1 this remain the same step 1 model 1 in this case in the model tree click on the job mode. So, definitely in the model tree we can click the job mode and name the job.

So, we can put the some job name, name the job and then click continue and then ok. So, then first click continue and then we are putting the name of the job name. And then we can put the in the toolbar click on the job manager. In the tool bar we can put the here, we can click the job manager.

(Refer Slide Time: 25:22)



Then, we can look into this also that once it is job 3 name this is particular job 3 model 1. And then we can choose the job type the full analysis, a complete analysis of the model and then and one continue and then we put the ok. Then, it is ready for the start of the click to submit and initiate the thermo mechanical analysis.

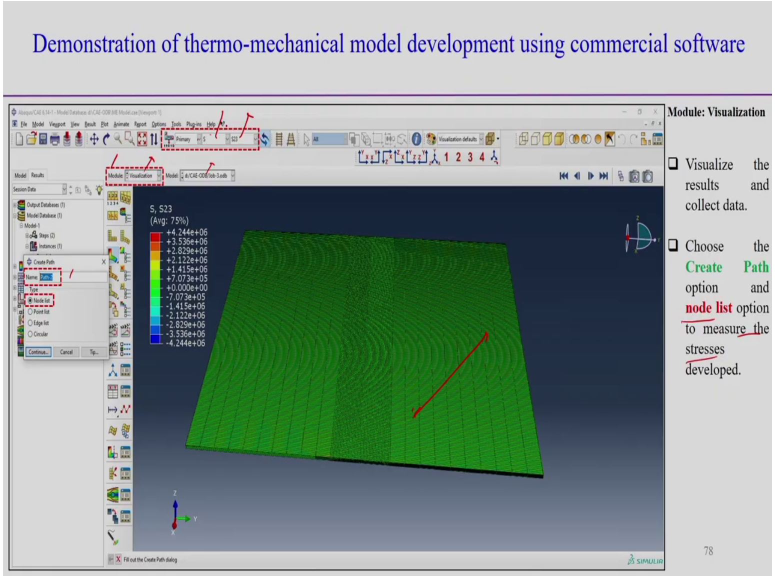
Once we put the ok then after naming the job name and the model, but after putting all the boundary condition and thermal load, then which define the job name then start the ok. And then it click the ok and it start the initially the start of the thermo mechanical analysis; that means, its continuing the thermo mechanical analysis starts.

So, therefore, once the thermo mechanical analysis starts, then it considers all the thermal load and the thermal load can be stored in the file as a function of temperature also. So, periodically it considered the or particular time step, it consider the thermal load and do the stress analysis model.

And, but the stress analysis is the same way what we have described theoretically that how stresses can be done. So, similar way the thermo elastic plastic analysis can be done also, using this particular finite element based commercial software. Now, once the stress analysis is done and then we can visualize the results.

So, therefore, the last module is the visualization of the results. So, visualization of the results here we can see the module we have to choose the visualization of the result. So, visualization of the results.

(Refer Slide Time: 26:46)



And here the model though we have considered the ODB file also. So, here we can see that visualization of the results and the we have to collect the data. Therefore, choose the create path option and node list option to measure the stress developed.

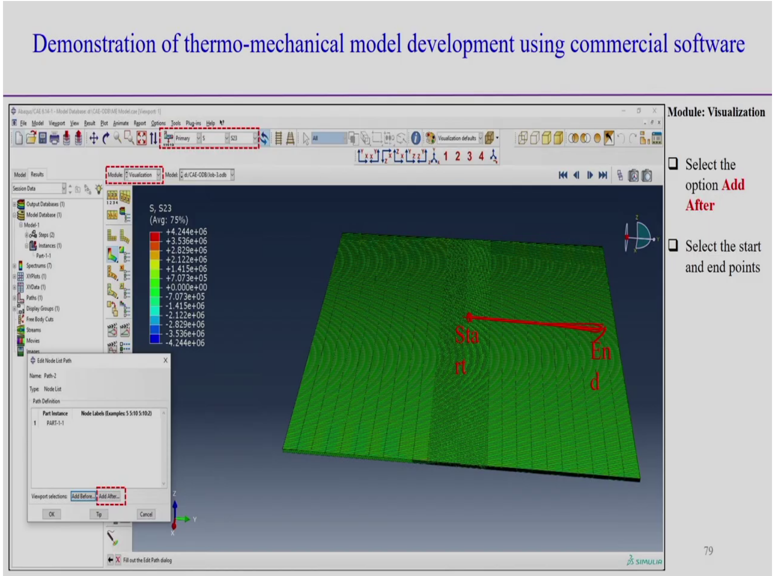
So, not only that the stress distribution for the whole domain apart from that if we want to know particular line, or particular node point the want to extract the data point also, or over a particular line from one point to particular point if you want to extract the data that also, also possible using this visualization mode.

Here you can see also that create path we can use a path to maybe in and type is the node list also. Node list in the sense that these properties or maybe thus parameters or stress distortion can be defined on the particular node; so, once we do these things and then choose the node list option to measure the stresses developed in during this process.

So, here also we can see that even if you see the create the this primary result and the S indicates the I think this is stress and S 23 is basically particular component so, then S 11, 22, 33, 12, 23. So, that indicates S 23 mostly the shear stress components or S 11, 22, 33 normally the normal stress components.

So, anyway if we choose the different values also here which results, which data we want to extract that we will be able to know from here that accordingly we will be getting the distribution of the particular stress value.

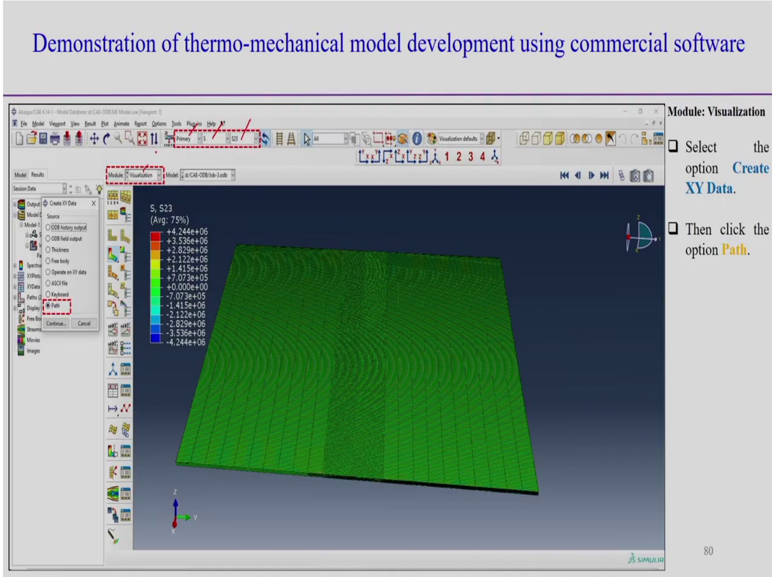
(Refer Slide Time: 28:17)



Now, visualization select the option add after. So, it is the start end and this is the start and other this is the end point. And select the start and end points. So, therefore, node levels and we will be able to know that we can add this path also to extract the data so; that means, select the option add after. So, if there is on add after if you put the view point selection so, that this the node list path.

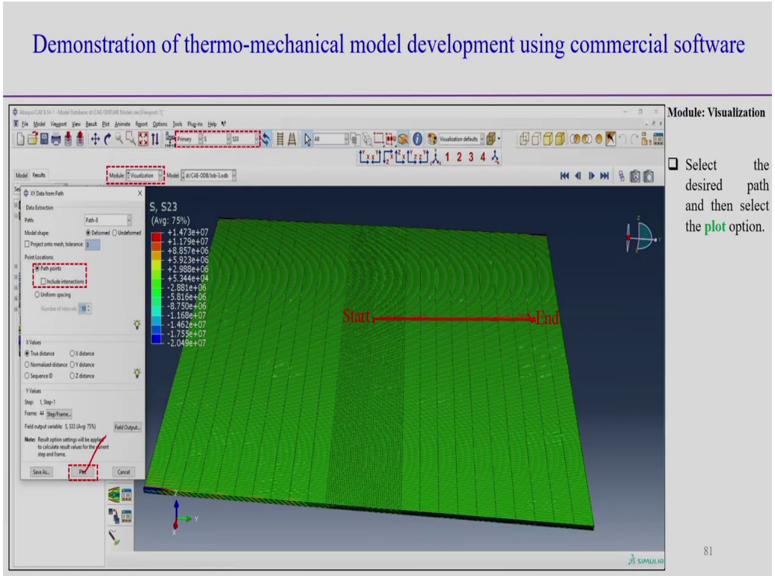
So, if path means suppose you want to know in this path what in the particular path what is the distribution of the particular component stress component value or distortion or whatever the value if we want to extract or if we want to see, then we have to choose this path, and then from here then here the if we add after that select the start and end points. Then, we can select the option add after then we choose the; we choose the start point and the end point to get the result between this line.

(Refer Slide Time: 29:12)



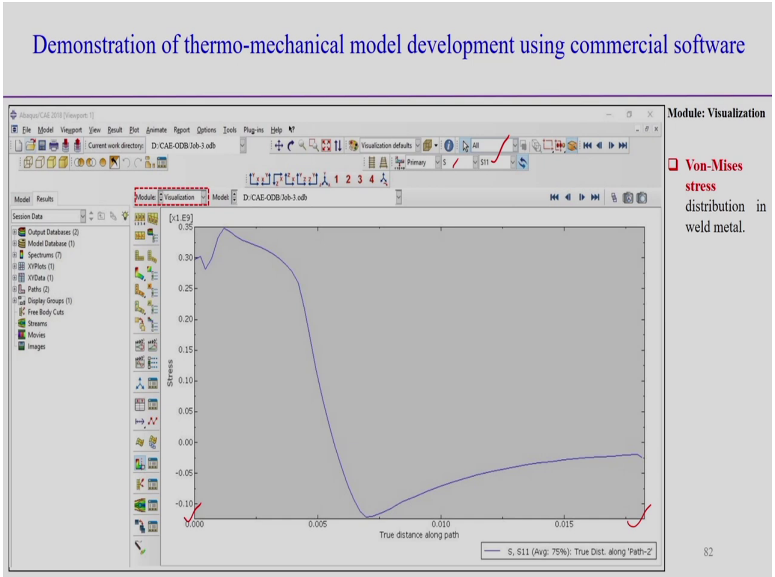
Now, select the option create XY data and then click on the option path also. And, then here we can create the XY data and then click on the option path, here we can choose the option path also. And, if we see that primary variable this stress and particular component of the stress and this module indicates the visualization of the result we can choose this path also path.

(Refer Slide Time: 29:34)



And here we can see the start and end point select the desired path and then select the plot option. So, select the desired path this is the start point, this is the end point and then we can see define this is a particular path name. And, then we can see the path points that include the intersection also. And then the design this is the true distance in the X value and distances, and then field output variable is the average value. And, then if you plot it if you choose the after choosing this path if you plot it plot option.

(Refer Slide Time: 30:06)

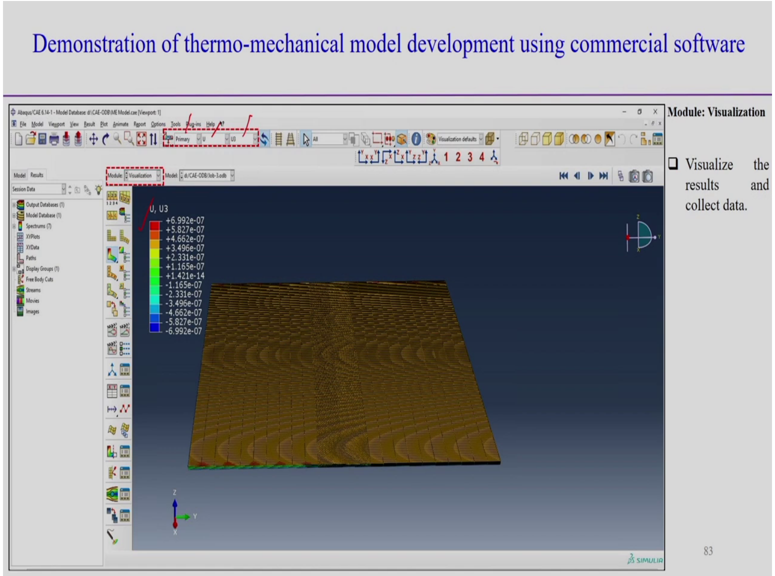


And then what we kind of result, we can get if we choose the plot we can see this is the results, we can see even see the true distance along the path. So, along the path true distance that the whatever the distance this is the reference as 0 distance and the these things the end point.

So, this is basically start point and this is the end point over this point, one particular component of the stress maybe in these cases particular stress component that we have already shown we have to choose any stress test S. And, we can see the if we want to see S 11 along this path then this is a typical plot of the this the value of the stress along this particular path.

So, this way we can visualize the result even if we need along the particular path also, we will be able to extract the data. And this it is called the Von Mises stress distribution in weld metal. So, one particular component the Von Mises stress distribution in the weld metal that we will be able to know also from this particular module.

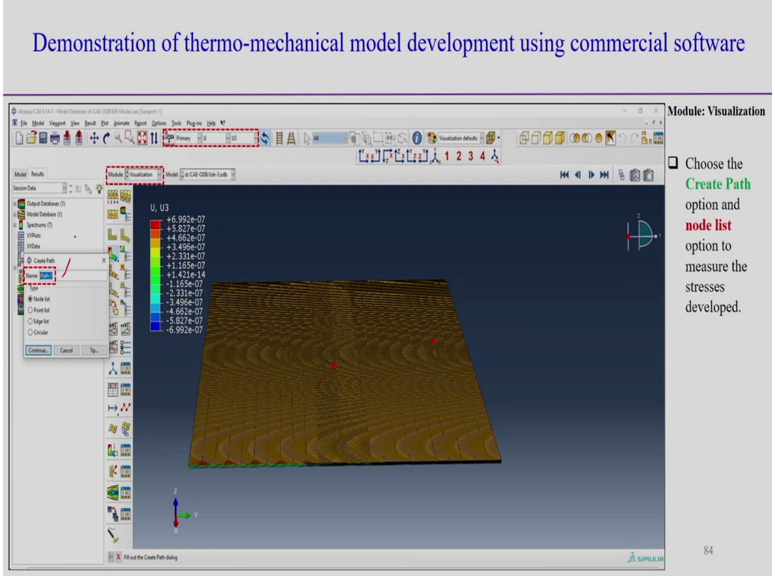
(Refer Slide Time: 31:01)



Similarly, visualize the results and collect the data also, here we can see visualize the result and collect the data in the different way we also see in these cases we have a primary variably the U, U will represent the displacement field, your distortion field also. So, here we can use the U 3; basically Z component of the distortion Z component of the distortion, we can put also and we will be able to know what is the Z component distortion.

And always we can see that one way we extract the result, this is scale bar the scale bar is color bar is also there the color bar indicates the which color represent what kind of the result the range of the distortion field, or may be any kind of the result in stress field that can be indicated by the color scheme. So, from the color scheme also we will be able to visualize the result during this analysis.

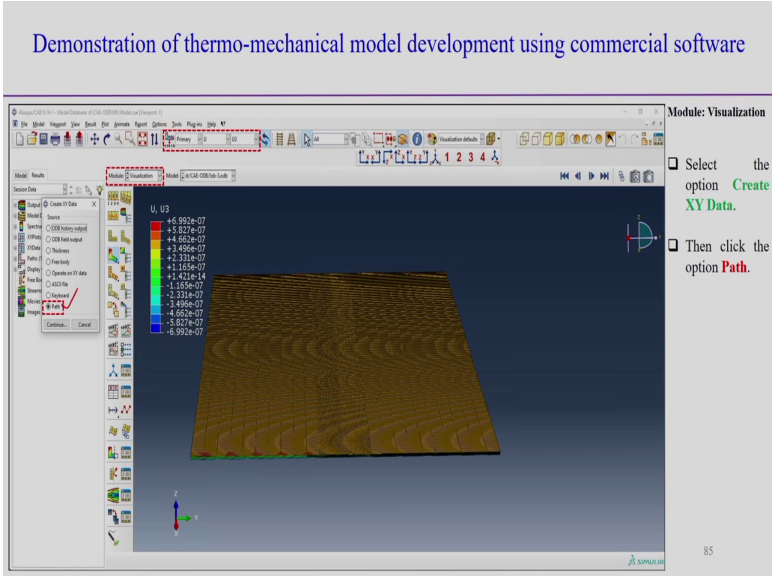
(Refer Slide Time: 31:47)



Now, here also we can choose the create path option and node list option to measure the stress developed, same way when they choose the we have to see the results over a particular path length. Then, you have to choose the path or we can choose the this particular node point and other node point between this node point what is the variation of the stress that can also be measured.

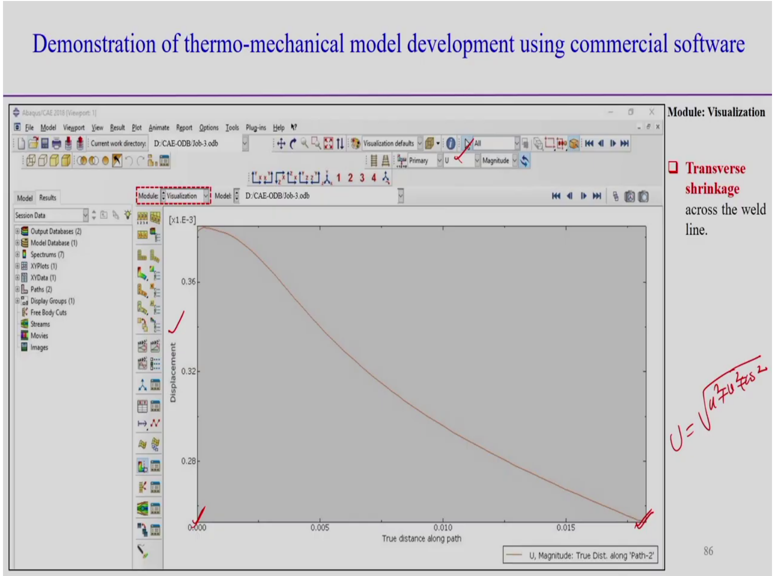
Stress development the similar way what we have done in case of the stress analysis, in the same way you can able to know the distortion field also for example, name equal to path 1 node list continue the same way.

(Refer Slide Time: 32:20)



Then we select the option create XY data, then we create the path option after that and once we create the path option, then click the option path, then we will be able to create the result also.

(Refer Slide Time: 32:33)



So, once you click the option path we can see that along this is the length selected length the initial point and the start point and the final point over this distance, what is the variation of particular variable? Here, in this case the particular variable the displacement field and displacement field also, and we can see the displacement U, U displacement here you can see the U primary and magnitude.

So, in these cases, I think we represent the U is basically all the component of the u square v square plus w square. So, that is the representation of this value variation along the distance. So, we can see that along the particular path we will be able to know the variation.

So, same this way we can visualize the results whatever form, it is required accordingly we can see all the results either or the whole domain the distribution can be plotted. Or are even not even if we choose the particular path also from one node point to another point, or path linear path if we choose it between these two points, it is also possible to find the distribution of the any particular variable.

For example, it can be only the single component of the stress or equivalence stress value, or may be single component of the distortion this U, the magnitude we consider, the resultant of the all the three components of the displacement field or individual displacement field. For example, either X, Y and Z components can also be obtained, can also be created this thing, can also be obtained from this particular software.

So, I hope this the demonstration of this thermo mechanical model. And in case of the stress analysis using some commercial software will be helpful, to start with working with this development of the thermo mechanical model using any kind of the finite element base commercial software.

So, that is all today thank you very much for your kind attention. And this is the end of the module 6 and maybe we will consider the next we will discuss the module 7.

Thank you very much.